

ECEN 250 Lab 1- Resistive Circuit Simulation and Measurement

Name: **Brodric Young**

Purposes:

- Simulate a resistor circuit with a voltage source and a current source
- Construct a resistor circuit and record measurements
- Verify measurements agree with hand calculations

Procedure:

Part 1 - SPICE simulation

Simulate the following circuits and paste a screenshot of the results under each schematic:

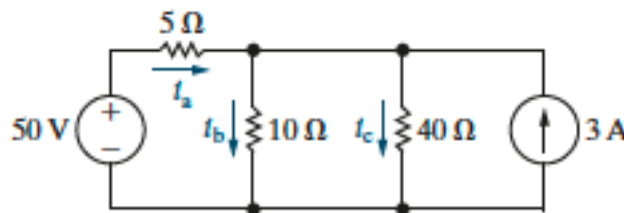
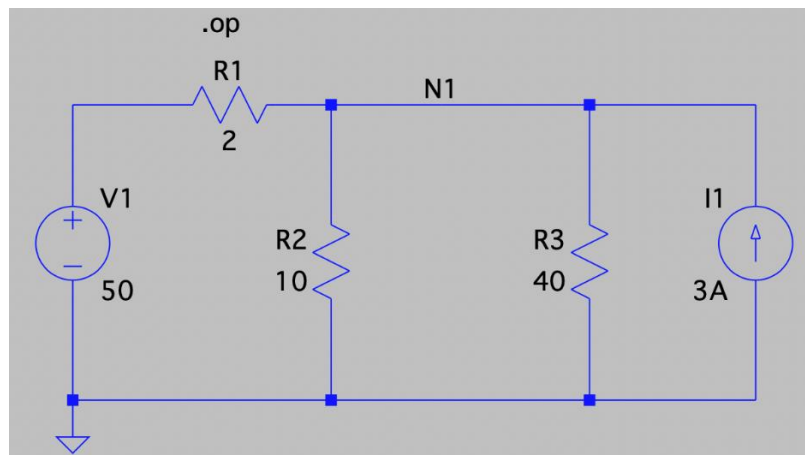



Figure 4.8 in the textbook (Example 4.3)



Example 4.3 - Simulating Voltage and Current Sources

Download the corresponding LTspice circuit file. **Double-check the resistor values in the schematic and change them if they don't match. 3**

Run the simulation using the  button.

If an operating point window pops up, take a screen shot of the operating points which includes the voltage at N1 (this is v_1 in the original problem), and the currents through each component. Insert it into this document.

If a plot pane pops up (instead of the operating point window) plot the voltage at N1 and the currents through the 5Ω , 10Ω , and 40Ω resistors (i_a , i_b , and i_c). To do this, right-click in the empty plot pane and select "Add Traces". All of the available voltages and currents are listed. Take a screenshot of this plot pane and insert it into this document.

Compare with the calculated results shown in Example 4.3 of the textbook:

	Calculated	Simulated
v_1	40V	40V
i_a	2A	2A
i_b	4A	4A
i_c	1A	1A

If the currents have the wrong sign, rotate the resistor in the schematic and re-simulate. To rotate, use the F7 function key to select the resistor, and use CNTL R to rotate.

```
* C:\Users\young\Downloads\Ex_4_3_DC.asc
--- Operating Point ---
V(n001):    50      voltage
V(n1):      40      voltage
I(I1):       3      device_current
I(R1):       2      device_current
I(R2):       4      device_current
I(R3):       1      device_current
I(V1):      -2      device_current
```

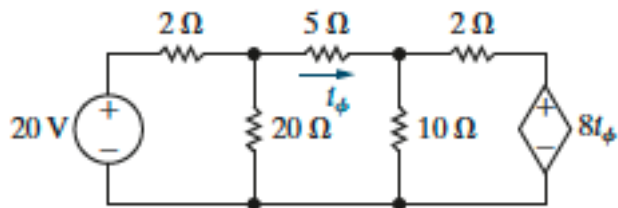
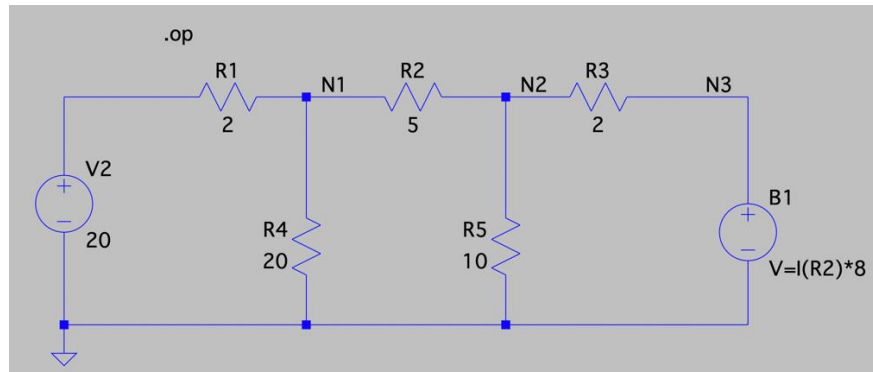


Figure 4.10 in textbook (Example 4.4)



Example 4.4 - Simulating a dependent voltage source

Take a screenshot of the operating points (as you did for Example 4.3) and insert in this document:

Compare with the calculated results shown in Example 4.4 of the textbook:

	Calculated	Simulated
v_1	16V	16V
v_2	10V	10V
i_ϕ	1.2A	1.2A

--- Operating Point ---

V(n001) :	20	voltage
V(n1) :	16.0001	voltage
V(n2) :	10.0003	voltage
V(n3) :	9.60049	voltage
I(B1) :	0.199919	device_current
I(R1) :	-1.99996	device_current
I(R2) :	1.19995	device_current
I(R3) :	-0.199919	device_current
I(R4) :	0.800004	device_current
I(R5) :	1.00003	device_current
I(V2) :	-1.99996	device_current

COMMANDS

SPICE Analysis	
.OP	find the DC operating point
.TRAN	perform nonlinear transient analysis
.AC	perform small signal AC analysis
.DC	perform DC source sweep analysis
.TF	find the DC small-signal transfer function
.NOISE	perform noise analysis

SPICE Directives	
.BACKANNO	annotate subcircuit pin names on port currents
.END	end of netlist
.ENDS	end of subcircuit definition
.FOUR	compute fourier component
.FUNC	user defined functions
.FERRET	download a file from URL
.GLOBAL	declare global nodes
.IC	set initial conditions
.INCLUDE	include file
.LIB	include library
.LOADBIAS	load a previously solved DC solution
.MACHINE	arbitrary state machine
.MEASURE	evaluate user-defined electrical quantities
.MODEL	define a SPICE model
.NET	compute network parameters in .AC analysis
.NODESET	supply hints for initial DC solution
.OPTIONS	set simulator options
.PARAM	user-defined parameters
.SAVE	limit the quantity of saved data
.SAVEBIAS	save operating point to disk
.STEP	parameter sweeps
.SUBCKT	define a subcircuit
.TEMP	temperature sweeps
.TEXT	user-defined string
.WAVE	write selected nodes to a .WAV file

SHORTCUTS

Schematic and Symbol Editing Modes		
Windows	Choose Mode then select component Exit mode: Press [Esc] or right-click	Apple
[F5] or [Delete] or [Ctrl]X	cut/delete	[F5]
[F6] or [Ctrl]C	copy/duplicate*	[F6]
[F7]	move* <i>unselected wires remain</i>	[F7]
[F8]	drag* <i>connected wires adjust</i>	[F8]
[Esc]	exit current mode <i>or right-click</i>	[Esc]

Zoom and Grid		
Windows	Zoom in and out with scroll wheel or track pad pinch	Apple
[Ctrl]Z	Schematic zoom area (drag over area) zoom in (click on scheme) Waveform zoom area is default mode [F9] for previous zoom Symbol zoom in	
[Ctrl]B	zoom out	
[Space]	zoom to fit (schematic viewer)	[Space]
[Ctrl]E	zoom extents (waveform viewer)	
[Ctrl]G	toggle grid	

TRICKS

Waveforms		
Windows	when clicking waveform label	Apple
click	add cursor and see measure	click
[Alt] click	highlight corresponding net in schematic	# click
[Ctrl] click	integrate waveform	[Ctrl] click

Schematics		
Windows		Apple
[Alt] click	component: plot instantaneous power wire: plot current	# click
hold [Ctrl]	draw wires at an angle	hold [Shift]
[Ctrl][Alt][Shift] H	show hidden component values/text, e.g. parallel or series resistance and capacitance	

any text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, active low, signal

Place Component Modes*		
Windows	Press [Esc] or right-click to exit place component mode	Apple
R	resistor	R
C	capacitor	C
L	inductor	L
D	diode	D
G	ground	G
V	voltage	V
S	spice directive <i>right-click text field to open "Help me Edit" dialog</i>	S
T	text/comment	T
[F2]	component	[F2]
[F3]	draw wire	[F3]
[F4]	label net	[F4]
	bus tap	B

*Rotate and Mirror		
Windows	*enabled in place modes	Apple
[Ctrl]R	rotate	# R
[Ctrl]E	mirror	# E

Undo/ Redo		
Windows	### Levels of Undo	Apple
[F9]	undo	[F9] or # Z
5 [F9] or [Ctrl]5 Z	redo	5 [F9] or # 5 Z

NUMBERS

Prefixes (Case Insensitive)		
LTspice	Means	Value
T or t	tera	10 ¹²
G or g	giga	10 ⁹
M or m	mega	10 ⁶
K or k	kilo	10 ³
M or m	milli	10 ⁻³
U or u	micro	10 ⁻⁶
N or n	nano	10 ⁻⁹
P or p	pico	10 ⁻¹²
F or f	femto	10 ⁻¹⁵

Constants	
LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0
mil	25.4×10 ⁻⁶ m

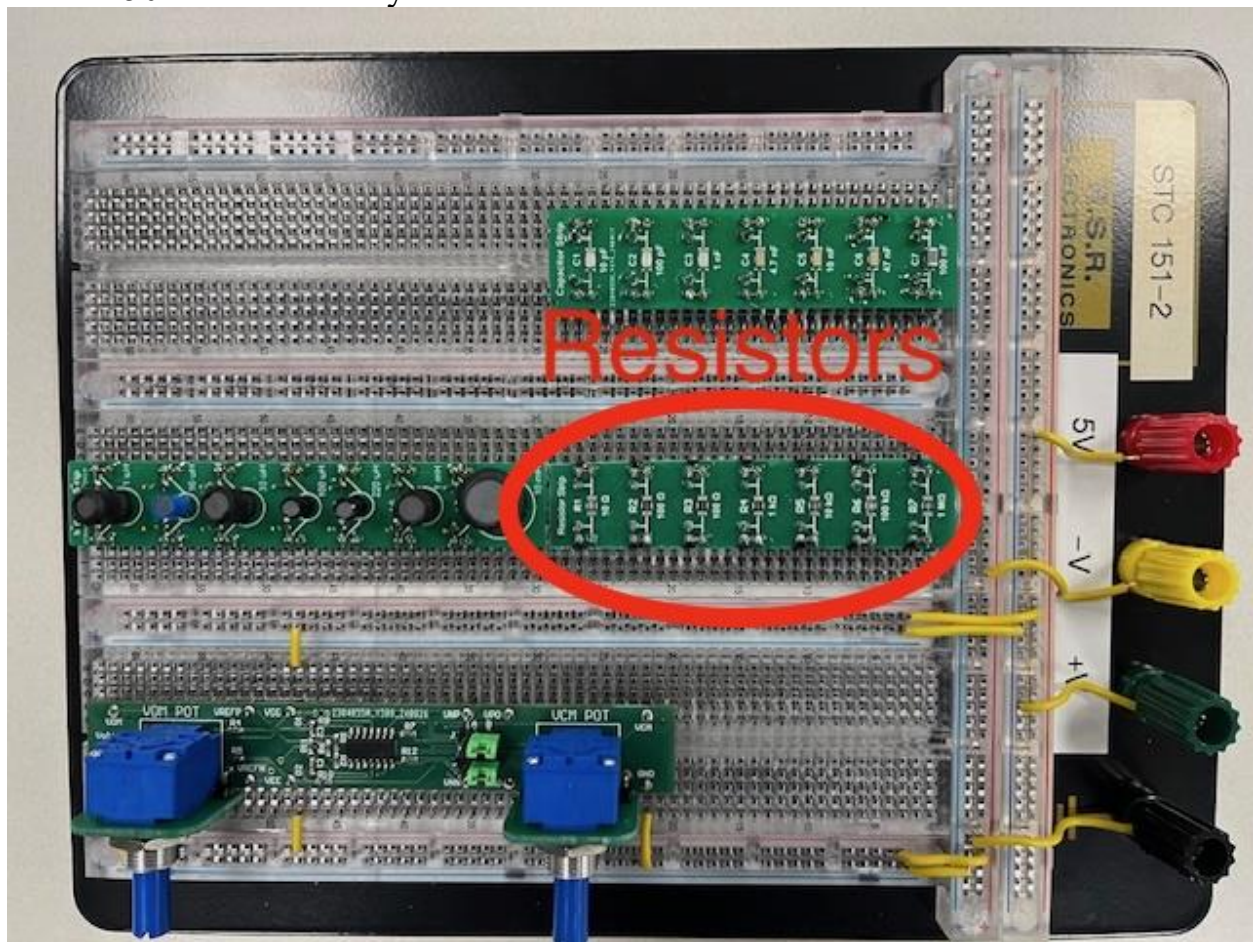


LTspice
Fast • Free • Unlimited

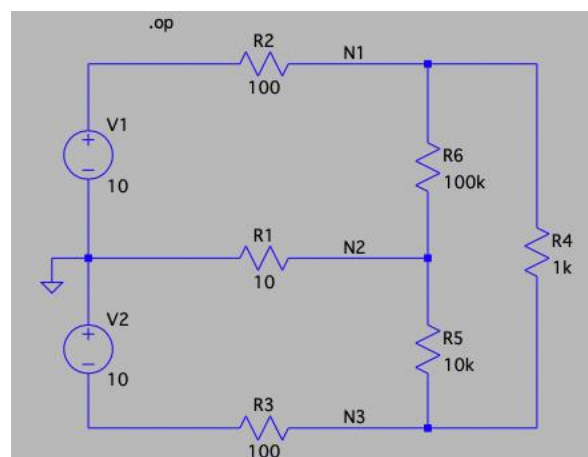
©2022 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners. Ahead of What's Possible is a trademark of Analog Devices. LTspice-4/22(A) analog.com

Part 2 - Circuit Construction

ECEN250 Bread-board System



Wire the following circuit:



Measure and record the voltages at N1, N2, and N3 (v_1 , v_2 , and v_3). All of these voltages are measured with respect to GND (0V).

	Measured	Calculated
v_1	8.70V	8.33V
v_2	-0.10V	-0.01
v_3	-8.12V	-8.26

Make your own mesh-current calculations and enter into table to compare with measured.

Conclusions (write a conclusion statement that discusses each of the purposes of the lab):

In this lab we used LTspice to simulate two different circuits, one with a voltage source and current source, and the other with a voltage source and a dependent voltage source. Using the simulations, it could calculate the voltages at different points in the circuit for us which was really nice. Then we physically built a third circuit and measured the voltage across different points similar to what were looking at in the simulations. We then verified that the measurements we made agreed with our own hand calculations of what it should be and everything checked out.